AIAA 97-0264
Numerical Investigation of Twin-Nozzle Rocket Plume Phenomenology

Houshang B. Ebrahimi
Sverdrup Technology, Inc., AEDC Group
Arnold Engineering Development Center
Arnold Air Force Base, TN 37389

Jay Levine
Phillips Laboratory/RKS
Edwards Air Force Base, CA 93524
and

Alan Kawasaki
Sparta Inc., Phillips Laboratory/RKS
Edwards Air Force Base, CA 93524

19980608 119

35th Aerospace Sciences Meeting and Exhibit
January 6-10, 1997 / Reno, NV

For permission to copy or republish, contact the American Institute of Aeronautics and Astronautics
1801 Alexander Bell Drive, Suite 500, Reston, VA 22091

DISTRIBUTION STATEMENT A
Approved for public release;
Distribution Unlimited

DTIC QUALITY INSPECTED 3

NUMERICAL INVESTIGATION OF TWIN-NOZZLE ROCKET PLUME PHENOMENOLOGY*

*Houshang B. Ebrahimi,*** *Jay Levine,*[†] *and Alan Kawasaki*[††]

Abstract

The Generalized Implicit Flow Solver (GIFS) computer program, developed under sponsorship of the Air Force Phillips Laboratory, Edwards Air Force Base, CA, has been modified and used for three-dimensional reacting two-phase flow problems. The intent of the original GIFS development effort was to provide the JANNAF community with a standard computational methodology to simulate multiple nozzle/plume flow-field phenomena and other three-dimensional effects. Recent development efforts have concentrated on improving the run time and robustness of the algorithm.

The GIFS computer program was originally released as an untested research version. Since that time, several corrections and enhancements have been made to the model. The Van Leer Flux Splitting option has been successfully implemented into the existing GIFS model and provides a more robust solution scheme. A Parabolized Navier-Stokes (PNS) version of the GIFS algorithm is currently under development and is intended to substantially improve the run-time requirements for flow fields dominated by supersonic flow regimes. These improvements and enhancements will foster the application of the GIFS model in the CFD community. This paper reports the significant results of several twin-nozzle/plume applications of the GIFS code. Six simulations of Titan II plume flow fields have been completed to assess the effects of three-dimensionality, turbulent viscosity, afterburning, near-field shock structure, finite-rate kinetic chemistry, intranozzle geometric spacing, and initial nozzle exit plane profile effects on the subsequent plume exhaust flow field. The results of these calculations indicate that the viscous stress model, kinetic chemistry, and nozzle exit profile are significant parameters that should be considered in analyses and interpretation of the calculations. In addition, the intranozzle geometric spacing influences the Mach reflection location which can significantly affect the plume/plume impingement shock location, inviscid shock structure, and shear layer growth.

Introduction

In order to support testing and analysis requirements of the plume community, a need exists for a fluid dynamics model which solves the fully coupled two-phase Navier-Stokes equations in multiple dimensions. Evaluation of solid-propellant rocket motor performance, nozzle erosion, and solid-propellant rocket plume radiative transfer analyses requires a computer model which simulates complex three-dimensional, chemically reacting two-phase flow effects.[1] Although this type of full Navier-Stokes method provides an accurate qualitative description of the basic features of the propulsion-generated flow fields, quantitative simulations for predicting fundamental parameters such as base pressure, static pressure, temperature, and chemical composition in the flow-field domain have not been validated. In the past few years, significant progress has been made in the areas of numerical rocket flow simulations and computational resources to the point that Navier-Stokes solutions are viable analysis tools.

The flow fields generated by rocket propulsion systems are complex, with regions of strong inviscid/viscous interactions, free-stream shear layers,

* The research reported herein was performed by the Arnold Engineering Development Center (AEDC), Air Force Materiel Command and was sponsored by Phillips Laboratory under contract No. F40600-95-C-0017. Work and analysis for this research were performed by personnel of Sverdrup Technology, Inc., AEDC Group, technical services contractor for AEDC. Further reproduction is authorized to satisfy needs of the U. S. Government.

** Sverdrup Technology, Inc./AEDC Group, Arnold Air Force Base, TN 37389-6001, Associate Fellow, AIAA.

† Phillips Laboratory/RKS, Edwards Air Force Base, CA 93524, Member, AIAA

†† Sparta Inc./Phillips Laboratory/RKS, Edwards Air Force Base, CA 93524, Senior Member, AIAA.

Approved for public release, distribution unlimited.

This paper is declared a work of the U. S. government and not subject to copyright protection in the United States.

nozzle wall and missile body boundary layers, external and internal shocks, separation regions, and plume/plume impingement and associated flow interactions for multiple nozzle designs, all with chemically reacting kinetics.[2] Modeling all of these phenomena simultaneously is a challenging task and is pushing the state of the art (SOA) for CFD models. To account for all phenomena affecting plume flow properties and the resulting radiative transfer implications, computationally efficient multidimensional computer models are required.

In the recent past, there has been increased emphasis on the application of existing CFD models, both in the commercial arena and government-developed computer programs, to simulate these complex multidimensional plume phenomena. Almost all of the solutions obtained to date have used the perfect gas approximation, as fully reacting flows are considerably more difficult to solve. Including the effects of chemistry in the solution produces a stiff set of equations which are numerically difficult to solve using conventional algorithms. In addition, the grid resolution requirements become more severe when reacting chemistry is included. When chemical reactions were first introduced into the GIFS Titan simulation, the original version of the GIFS code was unstable and did not produce a converged solution. This was a difficult issue to resolve and required a special enhancement to the GIFS algorithm.

Conventional rocket exhaust flow computer models[3,4] employ Euler solutions for the plume core flow and superimpose a turbulent parabolic mixing approach for the air/plume entrainment (shear) layer. These methods are generally not adequate for situations when the flow is not fully dominated by either inviscid core expansion or the plume afterburning phenomena. Three-dimensional features which result from multiple nozzle systems and vehicle/base interactions cannot be sufficiently treated through the use of these models. Inaccurate accounting of the upstream influences in the plume shear layer is one of the primary reasons which prevents the approximate models from accurate simulation of the overall flow field.[5]

The GIFS numerical algorithm provides a solution of the two- and three-dimensional Reynolds-averaged Navier-Stokes (NS) equations using the MacCormack implicit finite-volume algorithm with Gauss-Seidel line relaxation.[6] Several 2-D and 3-D plume flow-field calculations have been completed for the plume nearfield region using the original GIFS model.[6] The GIFS model includes a frozen and a generalized finite-rate kinetic chemistry model, a Lagrangian particle model for treating solid or liquid particulates, and a two-equation turbulence model, as well as a laminar model. These complex phenomena are required to accurately simulate the physics expected to contribute to the subsequent plume signature. The original objective of the GIFS development was to provide a flowfield model to be used in conjunction with radiative transfer models to predict base heating characteristics resulting from three-dimensional, multiple nozzle propulsion configurations. The Van Leer Flux Splitting option[7] has been successfully implemented into the existing GIFS model and provides a more robust solution scheme.[8] A Parabolized Navier-Stokes (PNS) version of the GIFS algorithm is currently under development and is intended to substantially improve the run-time requirements for flow fields dominated by supersonic flow regimes. These improvements and enhancements will foster the application of the GIFS model in the CFD community.

The plume structure of a multinozzle hypersonic vehicle flying at high altitude is largely dominated by the processes taking place in the plumes' interaction region. Prior to the release of the GIFS model, multiple nozzle/plume flow fields were commonly treated by assuming a single equivalent nozzle configuration having equal mass, energy and momentum of the multiple nozzle geometry. Further, a uniform nozzle exit flow was used as the starting conditions for the plume calculation. The simplified model assumes that the details of the 3-D flow structure in the near field flow are unimportant and that the flow processes which affect the plume shear layer initialization (such as base separation and recirculation) will be dominated by the overall ambient flow entrainment effects. The level of agreement between computations using the single equivalent nozzle methodology and radiometric data from multiple nozzle propulsion systems has not been acceptable, especially for spatial analysis requirements. The source of the disagreement is

DTIC QUALITY INSPECTED 3

due at least in part, to an incorrect physical model of the phenomena dominating the observations and can be further identified in the model as incorrect turbulence, chemistry, and an inadequate representation of the geometry.

The motivation for this study is to demonstrate the significance of three-dimensional effects as applied to multiple nozzle rocket missile plumes, to determine how these phenomena may be better simplified in order to promote improvements to engineering approaches, and to explore methods to reduce the overall CPU resource requirements for simulating three-dimensional solutions. The study presented in this paper represents the initial efforts in simulating the rocket exhaust flow field from a hypersonic vehicle in flight.

Computations

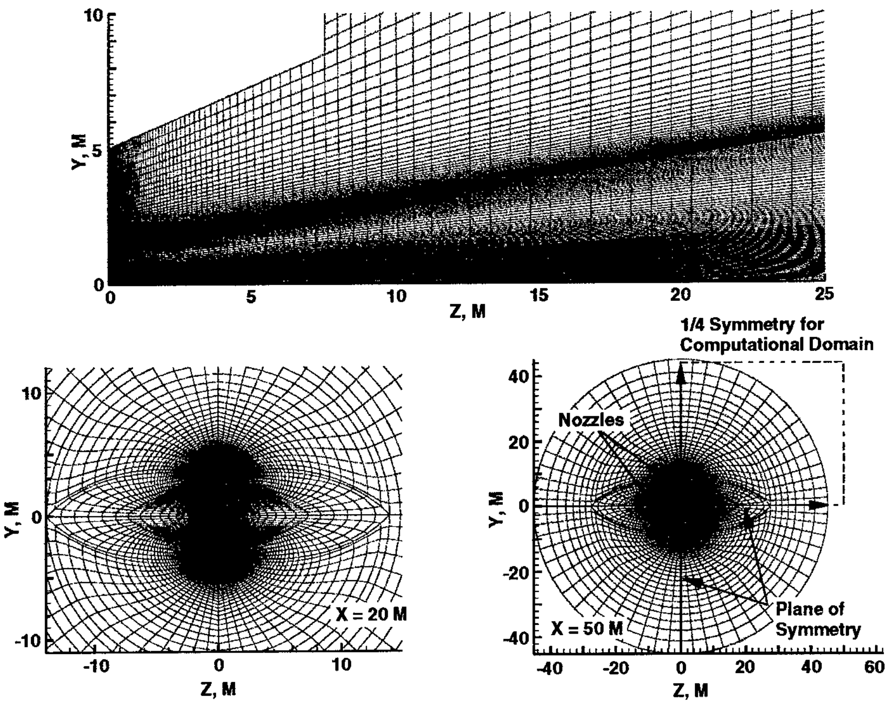
The computational effort consisted of six, three-dimensional twin-nozzle calculations for the Titan II vehicle at flight conditions. In all cases, only the plume flow field was computed, i.e., base effects were not considered. (Complete solutions including the flow over the missile body, the base region, and the exhaust plume domain will be presented in a future paper.) Also in the first five cases, boundary-layer effects from the rocket nozzle were not considered. The starting boundary conditions were assumed to be uniform. In the last case, the plume was computed utilizing two-dimensional radial profile starting conditions at the exit plane location of the nozzle. These were provided using the Two-Dimensional Kinetics Computer Program, TDK.[9] For all cases, the plume flow field was simulated at an altitude of approximately 50 km at a free-stream Mach number of 5.7. The resulting nozzle exit-to-free-stream velocity ratio was approximately 5, and the chamber pressure to freestream pressure ratio was approximately 50,000:1. The nozzle operated in all cases in a significantly underexpanded mode. The parametric flow-field conditions for all six cases are summarized in Table 1.

The Titan II Space-Launched Vehicle (SLV) propulsion system is propelled by two engines located on either side of a plane of symmetry passing through the vehicle centerline. To limit the number of grid points and CPU run time, quarter symmetry assumptions were made for the computational domain plane. Zero angle of attack and collinear plume exhaust were two additional constraints imposed for these simulations. A total of 600,000 grid points were utilized in the computational domain as shown in Fig. 1a with the orientation indicated by the axes. A schematic of the 3-D plume configuration is shown in Fig. 1b. The external airflow conditions and the liquid rocket nozzle exit conditions for the Titan test cases are presented in Table 2. The Titan three-dimensional calculations included finite-rate chemistry. For determining finite-rate chemistry effects, the chemical reaction model consisted of 10 species and 11 reactions to represent reactions for a carbon, hydrogen, oxygen and nitrogen-based propellant system. The κ-ε turbulence model was used for viscous stress approximation. The computation was performed on a single node of a four-processor SGI Power Challenge R8000. The calculation executed for approximately 11,000 iterations, and the most stressing case required approximately four weeks of CPU time to converge and complete.

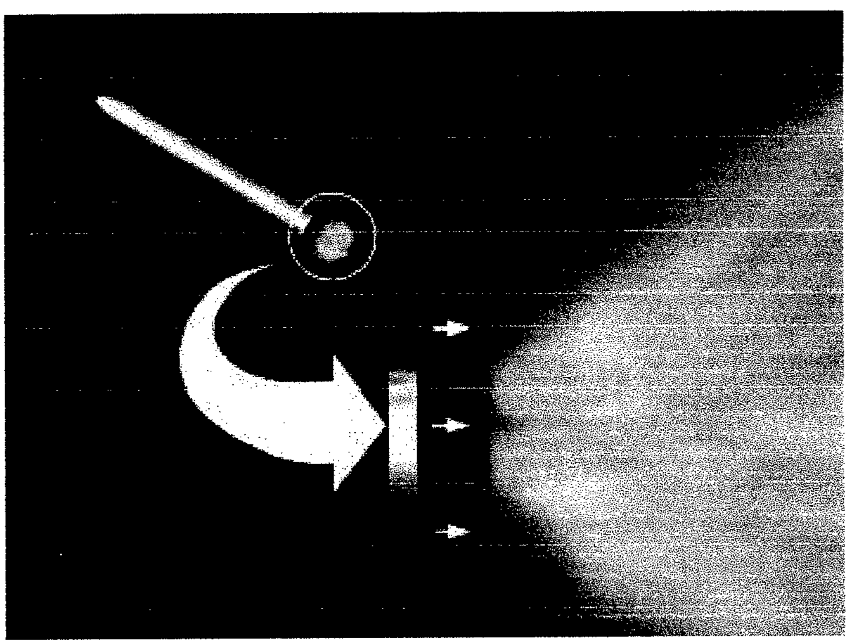
Six, three-dimensional numerical calculations of the twin-nozzle configuration were obtained at Mach 5.7 for different combinations of flow assumptions and approximations in an attempt to

Table 1. Parametric Flow-field Conditions

Case	Viscosity	Inlet Profile	Chemistry	Nozzle Spacing
1	Laminar	1-D	Constant γ	Narrow
2	Turbulent	1-D	Constant γ	Narrow
3	Turbulent	1-D	Constant γ	Wide
4	Turbulent	1-D	Finite Rate	Wide
5	Turbulent	1-D	Frozen	Wide
6	Turbulent	2-D	Finite Rate	Wide



a. Schematic of the 3-D grid and the planes of symmetry



a. Schematic of the 3-D plume
Fig. 1. Geometry representation of the computational domain.

isolate individual influences and effects. Cases 1 and 2 incorporated laminar and turbulent viscous stress models, respectively, and were computed assuming a perfect gas equation of state. Case 3 is a turbulent, constant gamma approximation with an intranozzle geometric spacing that differed from Cases 1 and 2. Cases 4 and 5 were obtained with frozen and finite-rate chemistry, respectively.

Cases 1-5 all assumed uniform nozzle exit properties as the GIFS start line conditions. The final case simulated a chemically reacting plume exhaust using radially varying profiles for the initial conditions at the nozzle exit plane (calculated via TDK) to define the starting boundary condition for the GIFS plume calculation.

The following sections will discuss the individual influences of the various assumptions for phenomena simulated in the GIFS model.

Three-Dimensional Effect

In an earlier study by one of the authors[2] and others,[10] it was shown that three-dimensional effects are important and should not be ignored or oversimplified in modeling efforts. In Ref. 2, a comparison of the two-dimensional axisymmetric solution with the three-dimensional twin-nozzle solution was accomplished to determine the impact of the single equivalent nozzle assumption. Figure 2 is a plot of static pressure contours for the three-dimensional solution. The plume expansion shock, barrel shock, and shock reflection are clearly visible at 116 meters downstream of the nozzle exit plane location. The static temperature is increased at the reflection point to approximately half the value of the total temperature of the flow. A single equivalent nozzle assumption provides valuable insight concerning overall gross qualitative assessments; however, for detailed studies requiring accurate resolution of the spatial character of multi nozzle plume flows, a three-dimensional calculation is required.

Turbulence Effect

In order to assess the influence of turbulence in the plume flow-field solution resulting from twin nozzles, two, three-dimensional calculations were performed: one assuming turbulent flow and the other assuming laminar flow. As would be expected, comparisons of the flow-field results for the turbulent and laminar cases indicates decreasing trends in plume impingement intensity, plume mixing rates, and a smaller plume cross section for the laminar calculation.

The comparison of static pressure contours in the vertical, horizontal and cross-planes is shown in Fig. 3. The turbulent and the laminar solutions are displayed separated centerline and indicated. Static temperature centerline axial profiles contrasting the turbulent and laminar solutions are

Table 2. Inflow Conditions

Ambient Conditions at 47.6 km			
- Tinf = 269 K	V = 1877.6 m/sec		
- Cp/Cv = 1.4	$\rho = 1.388 \times 10^{-3}$ kg/m^3		
- P = 108 Pa	Mach = 5.7		
- Species Concentrations (Mass Fraction)			
- N_2 = 0.77	O_2 = 0.23		
Jet Conditions			
- T = 1920 K	V = 2776.6 m/s		
- P = 92800 Pa	$\rho = 1.68 \times 10^{-1}$ kg/m^3		
- Mach = 3.0			
- Species Concentrations (Mass Fraction)			
CO = 0.039	CO_2 = 0.1811	H_2O=0.3496	
N_2 = 0.414	NO = 0.0109	OH = 2.139e-3	
H_2 = 3.13e-3	H =1.24e-4	O_2 = 0.0	O = 0.0

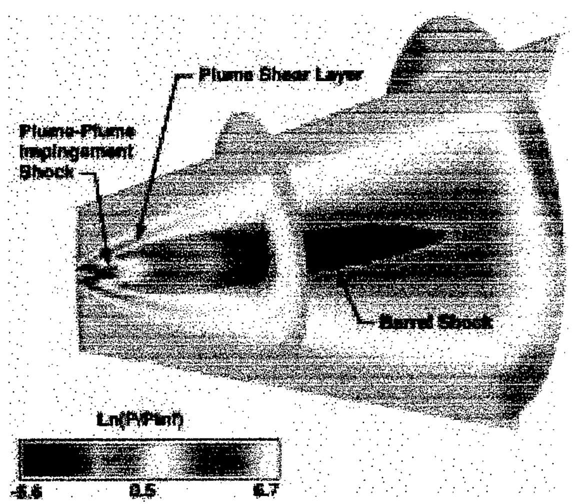


Fig. 2. Pressure contours (3-D calculation).

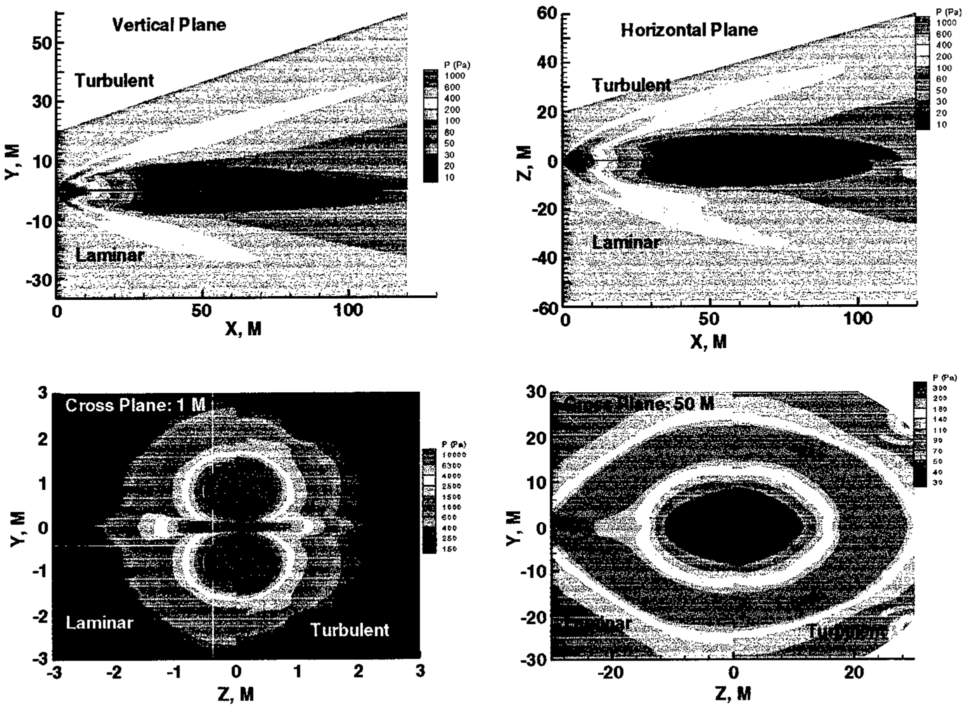


Fig. 3. Laminar and turbulent pressure comparisons.

shown in Fig. 4. These results indicate that the barrel shock reflection point is located approximately 8 meters farther downstream for the turbulent solution. Therefore, the spatial characteristics of the plume flow field and, hence, the location of the radiation centroid can be influenced by the viscous stress model included in the solution. In the present case at 47.6 km altitude, the effect of turbulence is not particularly strong. However, at lower altitudes and, hence, higher Reynolds numbers, turbulent effects will be more significant.

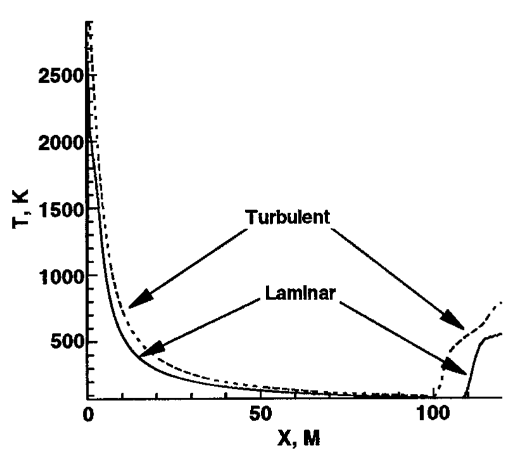


Fig. 4. Static temperature comparison along centerline.

Intranozzle Geometric Spacing Effect

Changing the spacing between nozzles should change the location and strength of the plume/plume interactions. In order to explore this effect, perfect gas, three-dimensional calculations were obtained at two different intranozzle geometric spacings (narrow versus wide spacing). The variation in the distance between nozzle centerline locations for the two cases was approximately 15 percent, with the first case assuming the widest separation distance. Figure 5 compares static temperature contours from the two solutions. These results indicate that intranozzle spacing has a noticeable influence on the flow-field structure. As seen in Fig. 5, the initial plume expansion angle is larger for the wider spacing case and the shock reflection location is farther downstream. A centerline axial profile of static temperature is displayed in Fig. 6, contrasting the two nozzle spacing solutions. The difference in the location of the shock reflect point is evident. Figure 6 also confirms that the increased static temperature of the wide spacing case

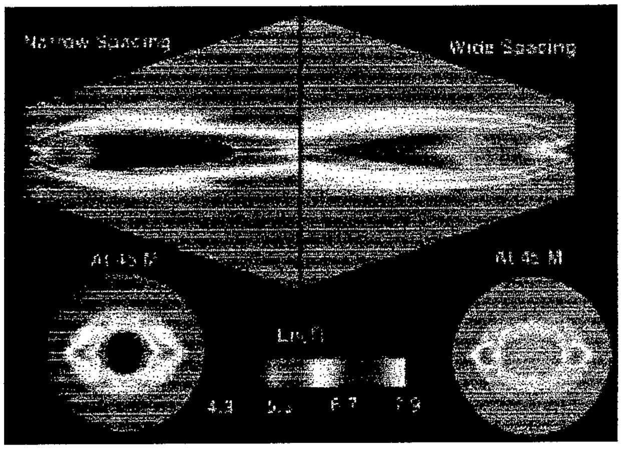


Fig. 5. Temperature contour of perfect gas comparison nozzle spacing effect.

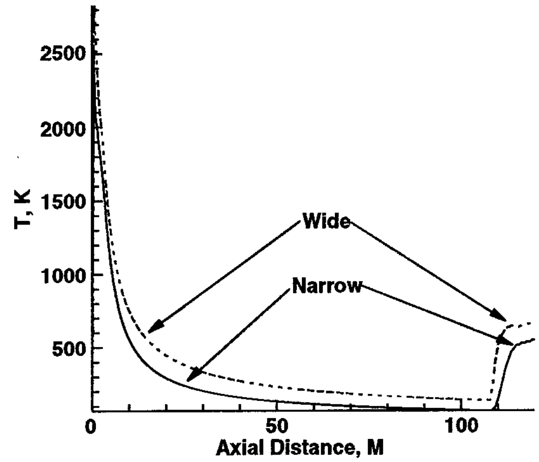


Fig. 6. X-Y plane comparing nozzle spacing effect.

extends throughout the calculation domain. It is concluded that the intranozzle spacing affects the plume impingement shock location, the inviscid shock structure, and the shear layer. Thus, nozzle spacing can significantly impact the flow field and is likely to influence the plume emission.

Kinetic Chemistry Effect

A frozen solution was contrasted with a finite-rate reacting solution to assess the significance of chemistry in complex plume flow fields. A comparison of the two calculations (Figs. 7 - 8, respectively) indicates that there is an effect on overall plume structure due to the chemistry model. The increase in static temperature attributed to the reflected shock is notably different between the solutions. The barrel shock reflection for the reacting flow solution occurs farther upstream than the frozen case and results in a strong normal shock Mach reflection/disc behavior. Further, chemical reactions occurring in the plume impingement and shock reflection region result in dissociation of the chemical species and also recombination combustion due to mixing with the air in the region between the nozzles. This phenomenon, which was not observed even in a finite-rate reacting single equivalent nozzle solution, is due to multiple plume interactions. Figure 7 compares temperature contours for the two cases; the frozen solution is displayed above the contour centerline and the kinetic solution below. The presence of compression-induced combustion in the downstream shock region is evident in the kinetic case. Figure 8 is a centerline axial profile of static temperature extending from the nozzle exit plane to 150 meters downstream. This comparison indicates that the chemis-

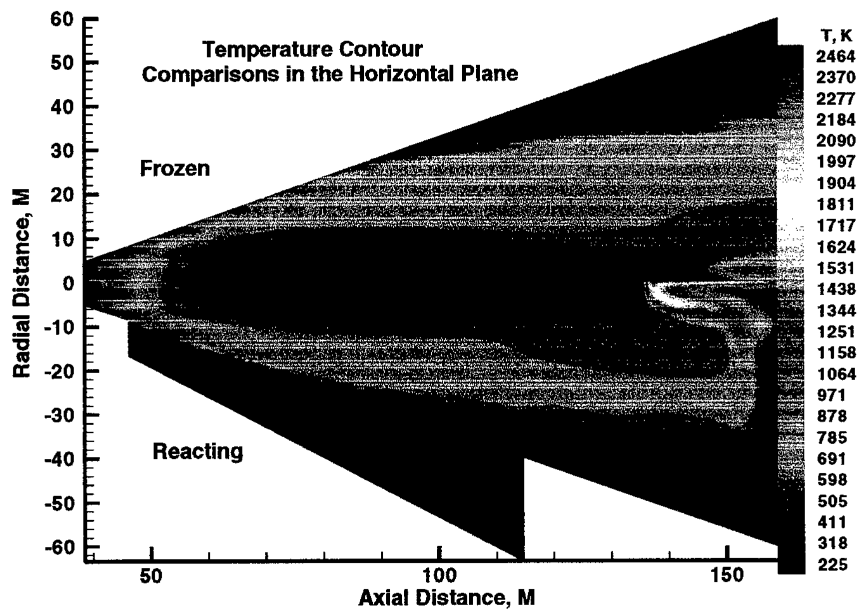


Fig. 7. Static temperature contours of reacting versus frozen flow.

try model has an effect on the location of the barrel shock reflection point. The reacting chemistry solution results in the shock reflection point occurring further upstream of the nozzle exit plane, and consequently, an overall stronger shock. The reacting chemistry centerline temperature reaches a maximum value of over 2,000 K behind the Mach disc (Fig. 8). For the frozen case, the corresponding maximum temperature is only 750 K.

Initial Boundary Conditon Profile Effect

In a previous study by one of the authors[5] to assess phenomena affecting scramjet nozzle performance, it was determined that performance is sensitive to variation in the radial profiles assumed for the initial boundary condition start line properties. Thus, the influence of the nozzle exit plane profile variations on the plume flow field was investigated. In one case, radial exit plane profiles were determined via the TDK model. In the second case, uniform exit plane profiles were assumed. Plume expansions were then calculated for each case assuming fully turbulent, chemically reacting flow. A comparison of static temperature contour plot and cross sections at axial positions of 1 meter and 7 meters downstream of the nozzle exit are shown in Fig. 9. The solution displayed above the centerline assumes a nonuniform start line profile. The solution below the centerline assumes a one-dimensional uniform start line condition. Radial profiles of static temperature are shown at axial positions equal to 3, 10, and 20 meters downstream of the nozzle exit in Fig.10. A comparison of the spatial plume size indicates that the uniform starting line results in a slightly larger plume expansion region due to the increased pressure gradient between the nozzle exit and the free stream. As expected, differences between the two start line approximations become less significant as the flow progresses axially downstream. Although these two solutions do not show a significant effect of exit profile shape on the plume tem-

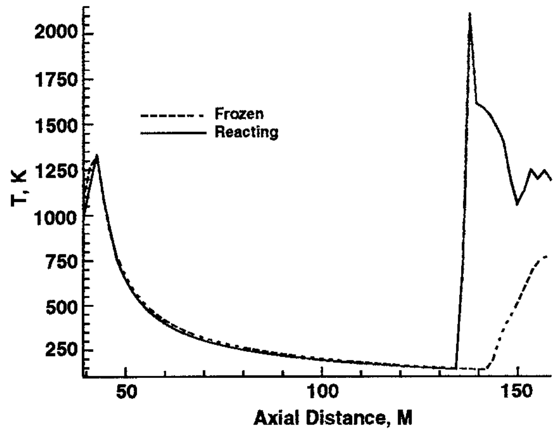


Fig. 8. Static temperature comparison of reacting versus frozen.

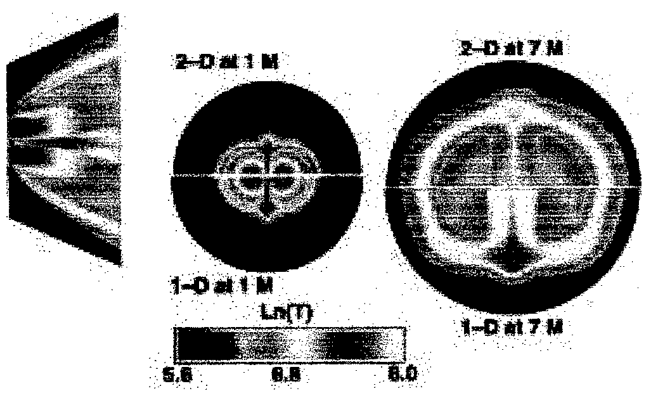


Fig. 9. Temperature contour comparison of 2-D versus 1-D start line.

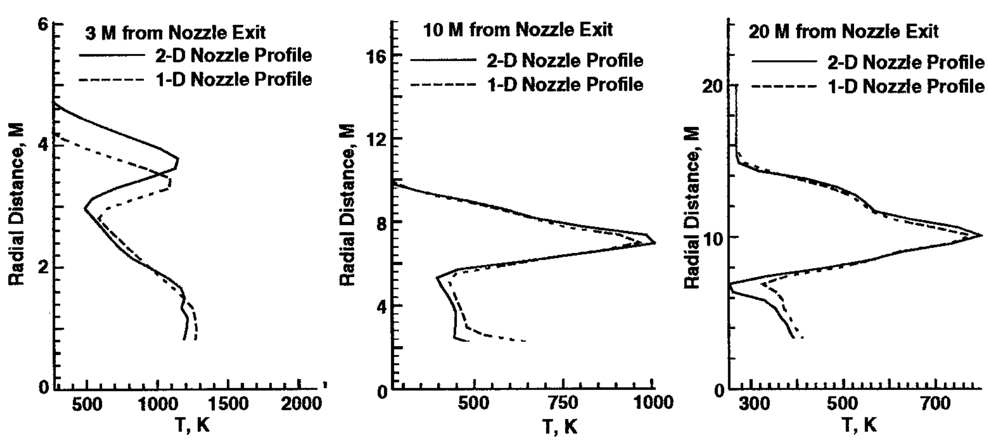


Fig. 10. X-y temperature line at X = 3, 10, and 20 m.

perature, it is expected that in other cases, especially those with strong afterburning, that larger effects would be noted. It should also be pointed out that the aforementioned solutions were obtained assuming constant oxidizer/fuel (O/F) ratio across the inflow plane. When real engine effects such as fuel-film cooling and injector imperfections are accounted for, O/F can vary widely across the nozzle exit plane. In order to properly account for the effect of such mixture ratio variations on the plume flow field and resulting base heat transfer effects, radially varying exit plane profiles must be employed.

Conclusions

The GIFS code has been significantly modified to improve the code execution, reliability, and user interface for three-dimensional flow-field calculations. A modified three-dimensional GIFS model was applied extensively for this study which included six different three-dimensional solutions. The effects of nozzle exit nonuniformity, intranozzle spacing, turbulence, and finite-rate chemistry were evaluated for a twin-nozzle/plume propulsion configuration. Analysis of the results considering these various effects leads to the following conclusions:

1. Nozzle exit start line assumptions, nonuniform versus uniform, had a significant effect on the immediate plume near field. The difference between the two start line assumptions became less significant as the axial distance downstream of the nozzle exit plane increased. This significance of the observed sensitivity needs additional investigation to elucidate the effect of the starting conditions on the plume flow field, especially in the presence of strong afterburning and missile body/base flow-field interactions.

2. Comparison of solution results contrasting laminar and turbulence stress models indicates that plume flow-field simulations are significantly influenced by the turbulence model. A laminar approximation is not appropriate for use in low- to moderate-altitude plume flow-field analysis.

3. The chemistry assumption (frozen versus reacting) did not influence the overall plume structure in this case. However, the chemically reacting solution produced a strong Mach reflection behavior and combustion in the vicinity of the Mach reflection zone which was not predicted by the frozen results.

4. The intranozzle spacing distance has a significant impact on the barrel shock reflection location, plume/plume impingement shock location, and shear layer.

5. A comparison of the two-dimensional and three-dimensional cases indicates that the three-dimensional effects are important in the near-field plume and diminish as the axial distance extends further downstream from the nozzle exit plane. The single, equivalent nozzle approach should not be used to describe plume near-field flow characteristics where three-dimensional effects predominate, and in the instances where 3-D features are required as part of the flow-field description.

6. These analyses indicate that two-dimensional nonequilibrium analysis tools can provide some insight concerning overall gross qualitative assessments of multiple plume flow-field phenomena. However, for detailed studies of complex flow-field phenomena, a more sophisticated three-dimensional calculation is required.

Furthermore, this study is intended to assess, understand, and quantify phenomena of the plume physics and to identify where simplified models are appropriate for use without introducing any significant error to the simulated flow field and conclusions which might be deduced from analysis of the flow-field simulation.

Future extensions to the study presented in this paper will explore three-dimensional vehicle/base interactions with the plume flow. Also, to promote optimal utilization of computer resources, hybrid Navier-Stokes/Parabolized Navier-Stokes methods are being investigated.

Acknowledgements

Acknowledgment is given to Mr. Roy Hilton for his valuable suggestions and technical support.

Special gratitude is extended to Ms. Martha A. Simmons for the wide range of assistance provided.

References

1. Shannan, Robert V. and Murray, Alvin, "Development of the KIVA-II CFD Code for Rocket Propulsion Applications," Tenth Workshop for CFD Applications in Rocket Propulsion, NASA Marshall Space Flight Center, April 28-30, 1992.

2. Ebrahimi, Houshang B., "Numerical Investigation on Multi-Plume Rocket Phenomenology," JANNAF, 1995.

3. Dash S. M., and Thorpe, R. D., "Shock Capturing Model for One and Two-Phase Supersonic Exhaust Flow," *AIAA Journal*, Vol. 19, July 1981, pp. 842-851.

4. Dash, S. M., Wolf, D. E. , Beddini, R. A. and Pergament , H. S., "Analysis of Two-Phase Flow Processes in Rocket Exhaust Plumes," *J. Spacecraft Rockets*, Vol. 22, No. 3, May-June 1985, pp. 367-380.

5. Ebrahimi, Houshang B., "Parametric Investigation of the Effect of Various Phenomena on the Performance of a Scramjet Nozzle," AIAA-95-6048, 6th International Aerospace Planes and Hypersonics, Chattanooga, TN, April 3-7, 1995.

6. Holcomb, J. Eric, "Development of an Adaptive Grid Navier-Stokes Analysis Method for Rocket Base Flows," AIAA-88-2905, Boston, MA, July 1988.

7. Van Leer, B., "Flux-Vector Splitting for the Euler Equations," in lecture note in *Physics*, Vol. 170, ISBN 0387-11948-5, Springer-Verlag, 1982.

8. Ebrahimi, Houshang B., "CFD Validation and Evaluation for Reacting Flows, Part III," AIAA-95-0735, Reno, NV, January 1995.

9. Nickerson, G. R., "Two-Dimensional Kinetics (TDK) Nozzle Performance Computer Program," NAS8-36863, March 1989.

10. Wrisdale, Ian E., Mikulskis, David F. and Gilmore, Martin R., "Navier-Stokes Calculations of a Twin Nozzle Rocket Plume," AIAA-96-1881, Thermophysics Conference, New Orleans, LA, June 17-20, 1996.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY *(Leave blank)*	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	January 1997	Technical Society Paper

4. TITLE AND SUBTITLE
Numerical Investigation of Twin-Nozzle Rocket Plume Phenomenology
AIAA Paper No. 97-0264

5. FUNDING NUMBERS
Job No. 0074

6. AUTHOR(S)
H. B. Ebrahimi, SvT; Jay Levine, PL-Edwards, and A. Kawasaki, Sparta, Inc./PL-Edwards

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
Arnold Engineering Development Center/DOT
Air Force Materiel Command
Arnold AFB, TN 37389-9011

8. PERFORMING ORGANIZATION REPORT NUMBER

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
Arnold Engineering Development Center/DOT
Air Force Materiel Command
Arnold AFB, TN 37389-9011

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES
Presented at 35th Aerospace Sciences Meeting and Exhibit in Reno, NV.

12a. DISTRIBUTION AVAILABILITY STATEMENT
Approved for public releae; distribution unlimited.

12b. DISTRIBUTION CODE
A

13. ABSTRACT *(Maximum 200 words)*
The Generalized Implicit Flow Solver (GIFS) computer program, developed under sponsorship of the Air Force Phillips Laboratory, Edwards Air Force Base, CA, has been modified and used for three-dimensional reacting two-phase flow problems. The intent of the original GIFS development effort was to provide the JANNAF community with a standard computational methodology to simulate multiple nozzle/plume flow-field phenomena and other three-dimensional effects. Recent development efforts have concentrated on improving the run time and robustness of the algorithm.

The GIFS computer program was originally released as an untested research version. Since that time, several corrections and enhancements have been made to the model. The Van Leer Flux Splitting option has been successfully implemented into the existing GIFS model and provides a more robust solution scheme. A Parabolized Navier-Stokes (PNS) version of the GIFS algorithm is currently under development and is intended to substantially improve the run-time requirements for flow fields dominated by supersonic flow regimes. These improvements and enhancements will foster the application of the GIFS model in the CFD community.

This paper reports the significant results of several twin-nozzle/plume applications of the GIFS code. Six simulations of Titan II plume flow fields have been completed to assess the effects of three-dimensionality, turbulent viscosity, afterburning, near-field shock structure, finite-rate kinetic chemistry, intranozzle geometric spacing, and initial nozzle exit plane profile effects on the subsequent plume exhaust flow field. The results of these calculations indicate that the viscous

14. SUBJECT TERMS
multi-plume, solid-propellant rocket motor, phenomenology, flow field

15. NUMBER OF PAGES
11

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	UL